Reprinted with permission from the March 2007 issue of…

Reducing the Cost of Misprinted PCBs

By Bill Schreiber, President, Smart Sonic Corporation, Canoga Park, CA

Cleaning of stencils and misprinted PCBs has taken an increasingly important role in surface mount technology. Fine and ultra-fine pitch lands, together with other advanced packages, place new demands on stencil cleaning. Paste volume is a critical issue for fine, ultra-fine, chip-scale, BGA and flip-chip components. Insufficient solder due to clogging of stencil apertures is a primary cause of defects. This results in poor paste printing and therefore; clean stencils are important in delivering the proper amount of paste. Process engineers estimate that approximately 70% of surface mount technology defects are due to solder paste printing problems.[1]

Misprinted PCBs are something all assemblers need to deal with sooner or later. If the misprints are discarded, production cost will rise (especially if the misprint is on side 'B' and side 'A' is already populated) and the assembler must deal with an additional hazardous waste stream. If attempts are made to recycle the board, cleaning decisions must be made: 1) Wipe the board and potentially contaminate vias and other difficult-to-clean areas. 2) Not wipe the board and potentially contaminate the filters and sumps of a batch or inline cleaner. 3) Possible degradation of OSP and other coatings. 4) If the board is double-sided and side A is already populated and side B is misprinted, the integrity of side A must be assured.

The cleaning fluid should be selected first based on soil compatibility. Poor solvency of the soil cannot be overcome by using mechanical force and, therefore the cleaning machine should be integrated with the cleaning fluid.[1]

The use of batch or inline cleaners designed for cleaning reflowed PCBs should be avoided as these machines are designed to manage only "fugitive" solder balls that have escaped the reflow process. Large quantities of raw solder paste will contaminate the filters, sumps and chamber walls of these machines, promoting the transfer of solder paste onto subsequent reflowed boards.

The stencil cleaner has been identified as a potential resource for cleaning misprints. However, stencil cleaners using spray-in-air technology share the same problems as spray board cleaners: Filters, chamber walls and plumbing quickly become contaminated with solder paste promoting the haphazard transfer of solder paste onto subsequent runs. The sprays will also broadcast the solder balls throughout the wash chamber causing them to become lodged under and around components located on the reflowed side of a double-sided board – a solder ball nightmare.[2] In addition, high pressure wash sprays or an air-knife blow-off for drying can bend delicate land bridges between apertures and thus be the root cause of a misprint problem.[1, 2]

Stencil cleaners using ultrasonic technology eliminate the concerns of solder ball contamination as the cleaning action of ultrasonics (cavitation) relies on gravity and the solder balls simply fall away from the contaminated board. Cleaning systems that use ultrasonic technology should orient the board with misprinted paste side down to allow gravity to carry the solder paste away from the reflowed side.[1]

Some assemblers my have reservations about employing ultrasonic technology for fear of potential component damage. This reluctance is due to fear of component damage that could come from use of a single-frequency ultrasonic agitation of cleaning agents, back when TO-5 and similar metal-packaged semiconductor packages were widely used. Much of this fear goes back 50 years.[3] 50 years ago, ultrasonic cleaning was a new and uncontrollable process and the assembles were cumbersome and delicate. Today; ultrasonics is the most controllable precision cleaning technology available. Coupled with the durability of today's SMT assemblies, the two technologies are very compatible.

Studies by GEC Marconi Ltd. and the Electronics Manufacturing Productivity Facility (www.empf.org) suggest an ultrasonic frequency of 40 kHz or higher is effective (the lower the ultrasonic frequency, the more aggressive the scrubbing action or cavitation) and should alternate or "sweep" +/- 3 kHz. Sweep technology eliminates the hot spots or focused ultrasonic energy in the wash bath.[1, 3]

Another important parameter is the power density. The power density is the amount of electrical energy delivered to the bath via the ultrasonic generators. This energy is measured in watts per liter of wash solution. For example, if the wash bath contained 100 liters and the generator is rated at 1000 watts, the power density would be 10 watts/liter. The power density should be around 10 watts per liter or less for safe cleaning of PCBs.[1]

Standard ultrasonic cleaning equipment should be avoided as these machines are often designed for industrial cleaning applications and employ the traditional approach of "the more power, the better" (15 – 25 watts per liter) without regard to potential damage caused to populated misprinted PCBs. Standard ultrasonic equipment must provide a higher ultrasonic power density to maintain the utility of various cleaning agents. Whereas an ultrasonic stencil cleaner is designed for a specific purpose and can utilize a specific chemistry. If the cleaning efficiency of the chemistry is high, low power density may be incorporated to promote safe cleaning of PCBs.

The stencil cleaning process can be instrumental in preventing or contributing to a misprint problem. A clean stencil simply provides a better print. If a stencil cleaner process is not cleaning efficiently, any amount of residual solder paste left on a stencil can dry hard as cement. If the stencil cleaner was not effective in cleaning the original fresh solder paste, it is unlikely to clean dry hardened solder paste. This hardened deposit will attract additional solder paste layers causing a "snowball" effect leading to blocked apertures and insufficient deposition of solder paste.

Additional information is available from the recently published Stencil and Misprinted Board Cleaning Handbook, a standard developed by IPC. This document is available as a free download at: www.SmartSonic.com/article.html or directly from www.ipc.org/onlinestore and search IPC-7526.

References:

1. A standard developed by IPC, Stencil and Misprinted Board Cleaning Handbook, IPC-7526, February 2007
2. Richard R. Clouthier, "SMT Stencil Printing: A Decision That Could Impact Production," Electronic Packaging & Production, July 1996
3. William Kenyon, "Why Not Ultrasonic Cleaning?" Surface Mount Technology (SMT), October 2004.